

# Fraction Step Method for Solving Incompressible Navier Stokes Equation

Devansh Purohit

B Tech (Aerospace), Amity University, Noida, Uttar Pradesh, India

## ABSTRACT

This paper contains the method used to solve Incompressible Navier Stokes (INS) equation. This study is conducted to see how the methods used in this paper gives a solution over a particular domain. Fraction Step Method is used for solving INS equation. Non-dimensional equation is taken for formulation. It is solved in two parts. With first part, intermediate velocity is calculated. Then next part is used to calculate pressure. Once pressure is obtained, then velocity is corrected for next time step. This is called Pressure correction method. This process is continued until steady state solution is obtained. After that, vorticity is calculated, and plotted. These plots are analyzed and compared with the experimental solution. Further a conclusion is made regarding whole paper.

**How to cite this paper:** Devansh Purohit "Fraction Step Method for Solving Incompressible Navier Stokes Equation"

Published in International Journal of Trend in Scientific Research and Development (ijtsrd), ISSN: 2456-6470, Volume-4 | Issue-5, August 2020, pp.193-196, URL: [www.ijtsrd.com/papers/ijtsrd31754.pdf](http://www.ijtsrd.com/papers/ijtsrd31754.pdf)



IJTSRD31754

Copyright © 2020 by author(s) and International Journal of Trend in Scientific Research and Development Journal. This is an Open Access article distributed under the terms of the Creative Commons Attribution License (CC BY 4.0) (<http://creativecommons.org/licenses/by/4.0>)



## INTRODUCTION

Property of a fluid element can be predicted for a domain by using the following three governing equation:

- Continuity Equation: Based on conservation of mass.
- Momentum Equation: Based on Newton's second law of motion.
- Energy Equation: Based on conservation of energy within a system.

These three governing equations were very accurate in determining the flow properties under a particular circumstance. Depending upon conditions, these equations are formulated for a specific problem. Like solving for compressible, incompressible, steady, unsteady, laminar, turbulent, isothermal flows. All the governing equations are partial differential equations therefore solved using numerical technique using power of computer.

Incompressible Navier Stokes equation is a case where spatial derivative for density is equal to zero. Thus, it gives a very simple equation for formulation. Non-dimensional form of Incompressible Navier Stokes equation will be used for the formulation:

$$\frac{\partial \mathbf{u}^*}{\partial t^*} + (\mathbf{u}^* \cdot \nabla^*) \mathbf{u}^* = -\nabla^* p^* + \frac{1}{Re} \nabla^{*2} \mathbf{u}^*$$

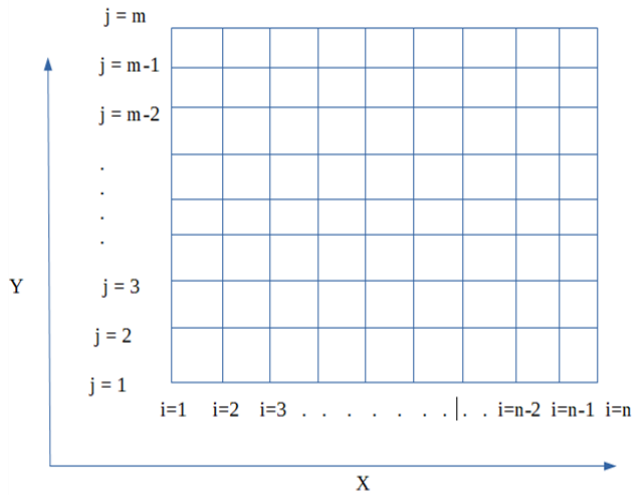
Here first term on LHS is the local derivative of velocity, second term is convective term for velocity, on RHS,

gradient of pressure is taken first and Laplacian of velocity, also called diffusion term. There is one more term which is ignored is gravitational acceleration term.

There is one approach for formulating Incompressible Navier Stokes Equation by formulating a general case for Compressible flows, and then put all spatial density derivatives as zero. But there is an error in this approach. For compressible flows, time step is calculated by the given formula:

$$\Delta t \leq \frac{1}{\frac{|u|}{\Delta x} + \frac{|v|}{\Delta y} + a \sqrt{\frac{1}{(\Delta x)^2} + \frac{1}{(\Delta y)^2}}}$$

For compressible flows, 'a' (speed of sound) is finite. So, it will give a finite value of time step. But for incompressible flows, the speed of sound is theoretically infinite. Which will give time step  $\Delta t = 0$ . This is not desirable as it not even start the calculation. It is also seen as when a compressible CFD solver is run for a Mach number near to zero, it takes more time steps to converge the solution. And most of the time it has tendency to be unstable at such low Mach numbers. Therefore, a different approach is used to make incompressible solver. Finite difference method is used for generating numerical solution of Incompressible Navier Stokes equation. FORTRAN Language will be used for the formulation of this solver.



**Figure 1: Grid used for formulation of Incompressible Navier Stokes Equation.**

Above figure contains a rectangular grid which represents the domain for this solver. All the small rectangular boxes are representing fluid element. Property of fluid element is calculated at the intersection point of two elements, also called as nodes. Length of a fluid element in X direction is  $\Delta x$  while the same in Y direction is  $\Delta y$ .

#### Fraction step method:

For solving the non-dimensional incompressible equation, fraction step method is used. For compressible flows pressure can be calculated using equation of state. But in incompressible form, equation of state cannot be used.

$$\frac{\partial u}{\partial t} + (u \cdot \nabla)u = -\nabla p + \frac{1}{Re} \nabla^2 u \quad (1)$$

For solving Incompressible Navier Stokes equation pressure gradient is needed. So, above equation is solved in two parts:

$$\frac{\partial \tilde{u}}{\partial t} + (u \cdot \nabla)u = \frac{1}{Re} \nabla^2 u \quad (2)$$

$$\frac{\partial u}{\partial t} = -\nabla p \quad (3)$$

Motive is to get velocity of fluid at next time step, so first velocity is calculated without taking the effect of pressure gradient. This velocity obtained is not the exact velocity for the fluid element at next time step. Therefore, this obtained velocity is called intermediate velocity ' $\tilde{u}$ '. This intermediate velocity is used in next equation to calculate pressure. Once pressure is calculated then with the help of pressure, velocity is corrected and new velocity is calculated for next time step.

#### Calculation of Intermediate Velocity( $\tilde{u}$ ):

For calculating intermediate velocity, eqn. 2 is used.

$$\frac{\partial \tilde{u}}{\partial t} + (u \cdot \nabla)u = \frac{1}{Re} \nabla^2 u$$

First term in LHS is the local derivative of velocity, next is convective term for velocity. The term on RHS is inverse of Reynolds Number multiplied by Laplacian of velocity vector.

Local velocity derivative is calculated with first order forward difference method:

$$\frac{\partial \tilde{u}}{\partial t} = \frac{\tilde{u} - u^{(n)}}{\Delta t}$$

here ' $n$ ' is the current time step.

For the convective term, upwind method is used. This is so because accuracy is not the priority but calculation time. Upwind method is therefore preferred for calculation of convection term.

Upwind scheme is:

$$(u \cdot \nabla)u = \frac{u_{(i)} - u_{(i-1)}}{\Delta x} \text{ if } u > 0$$

(or)

$$(u \cdot \nabla)u = \frac{u_{(i+1)} - u_{(i)}}{\Delta x} \text{ if } u < 0$$

If  $u$  which is outside of the bracket is positive then backward difference is used, if it is negative then forward difference is used.

For diffusive term on RHS, which is Laplacian of velocity vector, second order central difference is used:

$$\nabla^2 u = \left\{ \frac{u_{(i+1,j)} + u_{(i-1,j)} - 2u_{(i,j)}}{\Delta x^2} \right\} + \left\{ \frac{u_{(i,j+1)} + u_{(i,j-1)} - 2u_{(i,j)}}{\Delta y^2} \right\}$$

After arranging all the terms, intermediate velocity is calculated.

#### Calculation of Pressure:

Pressure for fluid elements is calculated using equation 3:

$$\frac{\partial u}{\partial t} = -\nabla p$$

First order finite difference is used for formulating local velocity derivative. For pressure term, central difference is used, because it converges and chances of instability is very least compared to forward and backward difference:

$$\frac{u_{(x)}^{(n+1)} - \tilde{u}_{(x)}}{\Delta t} = -\left( \frac{p_{(i+1,j)} - p_{(i-1,j)}}{2\Delta x} \right)$$

Above equation cannot be solved because there are two unknowns, one is ' $u^{(n+1)}$ ' and second is ' $p$ '. This can be solved with property of incompressible flows. It states that divergence of velocity is zero. If divergence of above equation is taken then ' $u^{(n+1)}$ ' will be reduced to zero. Intermediate velocity cannot be zero because it was not calculated taking the effect of pressure:

$$\frac{-\nabla \cdot \tilde{u}}{\Delta t} = -\nabla \cdot \nabla p$$

This gives a Pressure Poisson equation:

$$\nabla^2 p = \frac{\nabla \cdot \tilde{u}}{\Delta t}$$

Above equation contains Laplacian of pressure on LHS, which is equal to the source term on RHS. This source term is divergence of intermediate velocity over time step. This Pressure Poisson Equation is then solved in the given domain to obtain the value of pressure.

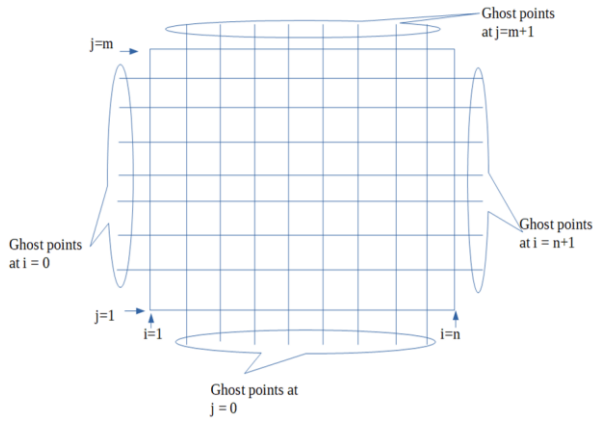
#### Solving Pressure Poisson Equation:

For solving Pressure Poisson Equation, it is converted into linear algebraic equation. For Laplacian of pressure, second order central difference is used:

$$\nabla^2 p = \left\{ \frac{p_{(i+1,j)} + p_{(i-1,j)} - 2p_{(i,j)}}{\Delta x^2} \right\} + \left\{ \frac{p_{(i,j+1)} + p_{(i,j-1)} - 2p_{(i,j)}}{\Delta y^2} \right\}$$

For source term, first order forward difference is used:

$$\nabla \cdot \tilde{u} = \left\{ \frac{\tilde{u}_{x(i+1,j)} - \tilde{u}_{x(i,j)}}{\Delta x} \right\} + \left\{ \frac{\tilde{u}_{y(i,j+1)} - \tilde{u}_{y(i,j)}}{\Delta y} \right\}$$



**Figure 2: Grid including ghost points for Boundary condition of Pressure Poisson Equation**

For starting the calculation boundary conditions are needed. Here Neumann boundary condition is used which states that normal component of pressure is equal to constant. Here the constant terms take is zero.

So:  $\frac{\partial p}{\partial x} = 0$  for  $x = 0, n$ ; and  $\frac{\partial p}{\partial y} = 0$  for  $y = 0, m$

First order central difference is used to formulate above conditions:

$$\frac{p(i+1,j) - p(i-1,j)}{2\Delta x} = 0 \text{ for } i = 1, n$$

And

$$\frac{p(i,j+1) - p(i,j-1)}{2\Delta y} = 0 \text{ for } j = 1, m$$

After getting pressure on boundary points, Pressure Poisson equation is solved on internal points:

$$p(i,j) = \frac{\left\{ \frac{p(i+1,j) + p(i-1,j)}{\Delta x^2} \right\} + \left\{ \frac{p(i,j+1) + p(i,j-1)}{\Delta y^2} \right\} - \frac{1}{\Delta t} \left\{ \left( \frac{\tilde{u}_x(i+1,j) - \tilde{u}_x(i,j)}{\Delta x} \right) + \left( \frac{\tilde{u}_y(i,j+1) - \tilde{u}_y(i,j)}{\Delta y} \right) \right\}}{2 \left( \frac{1}{\Delta x^2} + \frac{1}{\Delta y^2} \right)}$$

Using Gauss-Seidel method, whole Pressure Poisson Equation is iterated until solution converges. At last a set of values will be obtained which will be used to calculate velocity at next time step.

### Calculation of Velocity at next time step:

After getting pressure at all the points, the effect of pressure is used to calculate velocity for next time step:

$$\text{For X: } u_{(x)}^{(n+1)} = \tilde{u}_{(x)} - \Delta t \left( \frac{p(i+1,j) - p(i-1,j)}{2\Delta x} \right)$$

$$\text{For Y: } u_{(y)}^{(n+1)} = \tilde{u}_{(y)} - \Delta t \left( \frac{p(i,j+1) - p(i,j-1)}{2\Delta y} \right)$$

This method for using the pressure to correct and obtain next time step velocity is also called Pressure Correction Method. To obtain steady state solution, continuous iterations are done using obtained values until the solution converges for velocity.

### Result and Analysis:

After the solver is converged, a set of data is obtained from the solution. This data contains Pressure, X velocity and Y velocity. There is an additional step in the program, for analyzing the results, vorticity is calculated. This is done by:

$$\text{Vorticity} = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}$$

Above equation is formulated using first order finite difference method. The choice between forward and backward scheme is made depending on the boundary

Above conditions are used with ghost points and then substituted in Pressure Poisson Equation for each boundary condition. Ghost points are at  $j = 0, m+1$  and  $i = 0, n+1$ .

Above finite equations are simplified for boundary points:

$$\text{For } i = 1 \rightarrow p(2,j) = p(0,j);$$

$$\text{For } i = n \rightarrow p(n-1,j) = p(n+1,j)$$

$$\text{For } j = 1 \rightarrow p(i,2) = p(i,0)$$

$$\text{For } j = m \rightarrow p(i,m-1) = p(i,m+1)$$

Substituting these conditions in Pressure Poisson Equation, solution at boundary points are calculated by following equations:

$$p(1,j) = \frac{\left\{ \frac{2p(2,j)}{\Delta x^2} \right\} + \left\{ \frac{p(1,j+1) + p(1,j-1)}{\Delta y^2} \right\} - \left\{ \frac{1}{\Delta t} \nabla \cdot \tilde{u} \right\}}{2 \left( \frac{1}{\Delta x^2} + \frac{1}{\Delta y^2} \right)} \text{ for } i = 1$$

$$p(n,j) = \frac{\left\{ \frac{2p(n-1,j)}{\Delta x^2} \right\} + \left\{ \frac{p(n,j+1) + p(n,j-1)}{\Delta y^2} \right\} - \left\{ \frac{1}{\Delta t} \nabla \cdot \tilde{u} \right\}}{2 \left( \frac{1}{\Delta x^2} + \frac{1}{\Delta y^2} \right)} \text{ for } i = n$$

$$p(i,1) = \frac{\left\{ \frac{p(i+1,1) + p(i-1,1)}{\Delta x^2} \right\} + \left\{ \frac{2p(i,2)}{\Delta y^2} \right\} - \left\{ \frac{1}{\Delta t} \nabla \cdot \tilde{u} \right\}}{2 \left( \frac{1}{\Delta x^2} + \frac{1}{\Delta y^2} \right)} \text{ for } j = 1$$

$$p(i,m) = \frac{\left\{ \frac{p(i+1,m) + p(i-1,m)}{\Delta x^2} \right\} + \left\{ \frac{2p(i,m-1)}{\Delta y^2} \right\} - \left\{ \frac{1}{\Delta t} \nabla \cdot \tilde{u} \right\}}{2 \left( \frac{1}{\Delta x^2} + \frac{1}{\Delta y^2} \right)} \text{ for } j = m$$

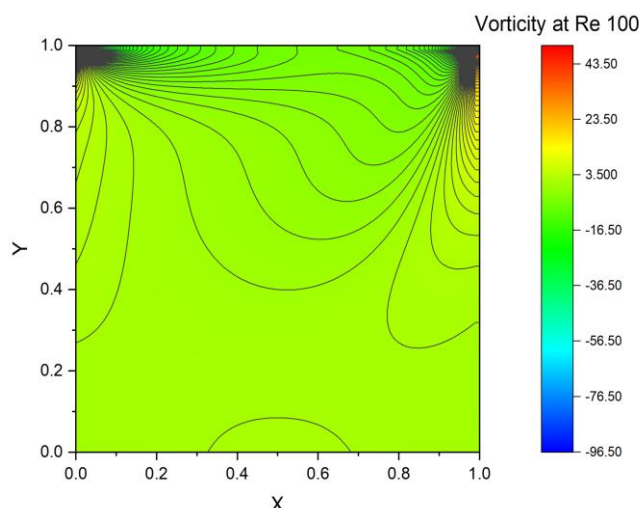
points. For all the points, forward difference is used but at  $i = n$  and  $j = m$ , backward difference is used.

For test case, condition of lid-driven cavity is simulated. Boundary condition for velocity is zero on all boundaries except at top boundary, where X velocity is taken as one. Test case is run for Reynolds Number equal to 100, 500 and 1000. A square grid of 100 by 100 is taken for test case. Results are obtained with a time step of 0.004 for all three conditions. Code was run was time period of 60 secs to get very accurate results and ensure steady state solution is obtained. This particular time step is chosen because it was the maximum value at which solution was stable. If time step is increased any further then there was instability in the code.

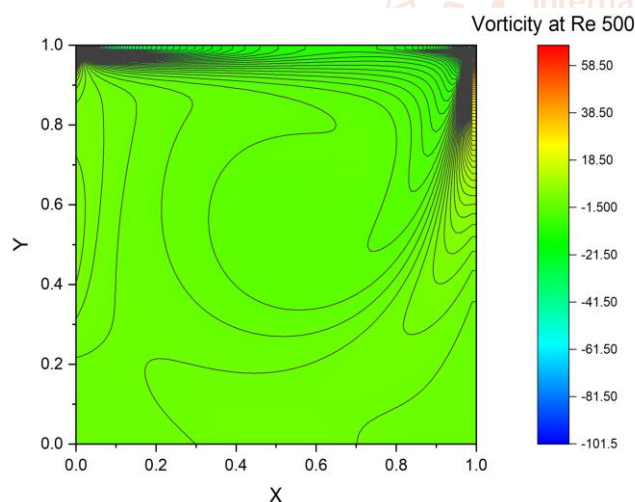
Results show that there is a significant variance from Reynolds Number 100 to 500. After that that change is not that prominent. For Reynolds Number 100: Formation of a thick boundary layer is seen across the side walls. The velocity at the bottom of the contour decreases significantly. Despite this, there are formation of strong vortices around the top corners of the domain. Vortices pass through the center for lower Reynolds Number. For Reynolds Number 500 to 1000: A significant change is



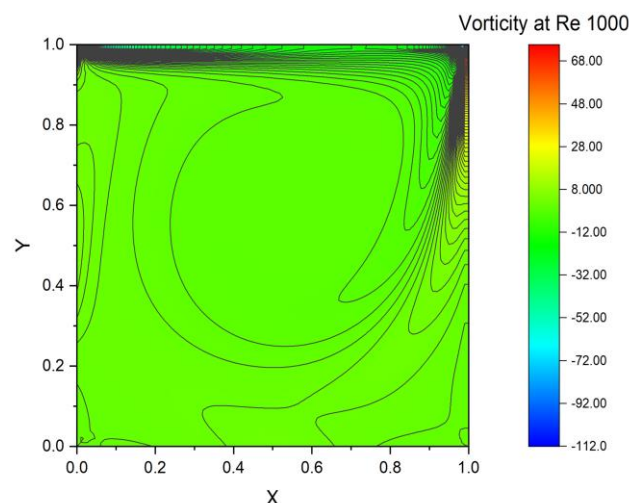
observed from  $Re = 100$ . Velocity is more around the center and top boundary of the domain compare to other regions. Unlike at  $Re = 100$ , where velocity gradually decreases as going to bottom boundary. It is also observed that as  $Re$  No. increases, a thin boundary layer forms at the side walls. Strong vortices are formed at the top corners, and here vortices do not pass through the center. Because velocity is more at the center, flow is circulating about the center of the domain. As  $Re$  No. increases, this circulation moves further away from the center. For  $Re = 1000$ , boundary layer is thinner, and there is formation of small vortices at the bottom.



**Figure 3: Vorticity Contour Plot for Reynolds no: 100**



**Figure 4: Vorticity Contour Plot for Reynolds no: 500**



**Figure 4: Vorticity Contour Plot for Reynolds no: 1000**

Center of the primary eddy remains constant below the top lid for Reynolds number 100. While for higher  $Re$  No. it lies at the center of the domain. One common observation for all the cases is boundary layer is thinner at top boundary because of  $X$  component of velocity there.

### Conclusion:

The solver for Incompressible Navier Stokes equation was successfully formulated. Use of finite difference was made and how different schemes affect the solution of the problem was observed. A small change in input parameters can result to instability in the solution and output inaccurate or no result. Formulation of Pressure Poisson Equation and boundary condition were seen as the most critical part of the solver. A very simple error of array dimension misplacing can diverge the solution. For different set of numerical schemes, different solutions are obtained. If forward or backward difference is used in pressure term while correcting velocity for next time step, then solution never converges and causes instability. This shows that proper set of schemes are used to give stable results. After obtaining solution, data were plotted in OriginLab pro software for visualization and analysis. It was seen how the dynamics of flow differs as Reynolds Number increases. Further then the obtained solution was compared with standard data available on internet and accuracy was obtained while obtaining the solution.

### REFERENCES

- [1] "Computational Fluid Dynamics" by "John D Anderson Jr."
- [2] "Fractional step methods for the Navier-Stokes equations on non-staggered grid" by "S. W. Armfield and Robert Street"